# Effect of Counter-Rotating Cylinder with Surface Roughness on Stagnation and Separation Point – A Computational Approach

### M. T. Abu Seman<sup>1\*</sup>, F. Ismail<sup>2</sup>, H. Yusoff<sup>3</sup> and M. A. Ismail<sup>2</sup>

<sup>1</sup>Department of Mechanical Polytechnic, Seberang Perai, Malaysia; mizie@psp.edu.my <sup>2</sup>School of Mechanical Engineering University, Sains - 11800, Malaysia <sup>3</sup>Faculty of Mechanical Engineering, University Teknologi MARA (Pulau Pinang), Malaysia

## Abstract

This paper is devoted to a two-dimensional numerical study of the influence of surface roughness on the cross-flow around a Counter-Rotating Cylinder. The surface roughness consists of irregular notches periodically distributed on the cylinder surface. The roughness was varied using different notch patterns and heights. The roughness patterns were tested on the cylinder. It was investigated approximately using two-dimensional block with average height and different shapes periodically distributed on a smooth cylinder. The influence of the roughness on the drag coefficient, stagnation point pattern and location of separation point was studied numerically over the range of Reynolds numbers 2.0 x  $103 < \text{Re} < 4.2 \times 104$  at rotating speed 400 rpm. In the Counter-Rotating Cylinder with roughness, no as well as different observed on drag coefficient prediction compared with previously published. However, the stagnation point and separation point on roughness cylinder is quite difference between a smooth cylinder.

Keywords: Circular Cylinder, Reynolds Number, Rotation Speed, Surface Roughness

# 1. Introduction

In recent years, Long-Span Suspension Bridges and Cable-Stayed Bridges have been widely constructed worldwide due to their superior structural performance and elegant appearance. As key components of Cable-Stayed Bridges, inclined cables often vibrate under Wind, Rain and Traffic loads, such as the Vortex-Induced Vibration (VIV)<sup>13,14,19,20</sup> and Rain-Wind Induced Vibration (RWIV)<sup>10</sup>. Although the VIV of the cable is a self-limiting vibration, it frequently occurs at lower wind speeds<sup>3</sup>. observed the VIVs of cables on the Fred Hartman Bridge in Houston, Texas, USA through a long full-scale measurement investigation. Frequent wind-induced vibration may induce fatigue damages to the cables, and mitigation is necessary to reduce the vibration.

Viscous damper devices have been widely used to control the VIVs and RWIVs of the stay cables of Cable-Stayed Bridges<sup>13</sup>, obtained an additional optimal damping ratio to the stay cable provided by the Viscous damper<sup>19</sup>,

\* Author for correspondence

employed an active stiffness control method to suppress the RWIVs of prototype stay cables<sup>12</sup>. Investigated the theory, algorithm and model test of the semi-active control of stay cables wind induced vibrations using magnet or heological fluid dampers, which can effectively reduce the cable vibrations and increase the stay cables system damping ratios.

The control mechanisms of each type of damper device provided additional damping ratios to the stay cables and increased their ability to resist wind-induced vibrations, but the methods did not change or reduce the wind loads on the stay cables. Aerodynamic methods, including passive and active approaches, can be used to reduce the wind loads on the stay cables. They can simultaneously increase the natural damping ratios of the stay cables at certain times<sup>2</sup> proposed the use of protuberances to suppress the RWIVs of stay cables. Gu<sup>9</sup> conducted wind tunnel tests using double-spiral wires to mitigate vibration<sup>15</sup> attached hemispherical bumps to the circular cylinder surface to control the vortex-induced vibration amplitudes of the cable models.

Many active aerodynamic approaches were also used to control the flow separation and flow-induced vibration. Wu et al<sup>18</sup> developed a moving-wall, e.g, appropriate travelling transverse wave, control strategy to manage the unsteady separated flow over a circular cylinder. This method allows the global flow to remain attached when there is a strong adverse pressure gradient and eliminates vortex shedding. Patnaik and Wei<sup>16</sup> investigated the use of momentum injection to control the flow field around the airfoils, flat plates, rectangular prisms, D-section prisms and circular cylinders; the authors found that this method could effectively resist the wind-induced vortex and galloping instabilities. Grager et al<sup>7</sup> used a dynamic burst control plate to suppress stall on an airfoil by preventing the low-reynolds-number leading edge separation bubble from bursting.

Suction flow controls were also widely used to suppress the flow separation of airfoils<sup>8,6,3</sup> and circular cylinders<sup>5,17</sup> by tests or CFD numerical simulations. In airfoils system, the moving surfaces are provided by rotating cylinders located at the leading edge or trailing edge as well as at the top surface of an airfoil. It has been shown that this mechanism of moving surfaces can prevent flow separation by retarding the initial growth of the boundary layer, with important consequences for lift enhancement and stall delay.

However, a circular cylinder has large dynamic drag resulting from the separation of flow over the cylinder. To reduce the drag coefficient of a circular cylinder, some methods have been studied, such as a Cylinder with Roughened Surface, and so on. However, it is not easy to obtain a roughened surface in the flow accumulate on the Cylinder Surface<sup>1</sup>. Beside of this scenario, the current work is to carry out well-resolved CFD simulations of the flow past two-dimensional rotating cylinder with roughness. The flow past a circular cylinder produces several flow features associated with more complex geometries. The geometries used in this work have difference of stagnation and separation point followed by rotating sped 400rpm and the range of Reynolds number 2.0 x  $10^3$  < Re < 4.2 x 10<sup>4</sup>. The focus is on identifying changes in stagnation point and separation flow in respect of maximum counterrotation and high roughness on cylinder.

# 2. Methodology

## 2.1 Roughness Profile

Figure 1 shows a cylinder with degrees of surface

roughness are prepared. To obtain a rough medium, the surface of the cylinder was covered with sandpaper. For a smooth medium, a cylinder was polished to a 5 micron surface finish such that the blockage and end effect (twodimensional) on the flow pattern will be minimum and it was set up in a wind tunnel. However, the roughness was obtained by placing various shaped irregularities at intermittent locations on a smooth Cylindrical Surface.



#### **Figure 1.** Cylinder with roughness.

An IF-Profiler (Alicona IF-Profiler) was used to measure the surface roughness of the sandpaper. The value of the surface roughness was dependent on the scale of measurement. Furthermore, the concept of roughness had statistical implications as it took into account such factors as sample size and sampling interval. Three measurements were taken at three different spots on each strip of sandpaper and the average of these three readings was taken as the roughness value for each strip. The IF-Profiler was used to determine the average surface roughness ( $R_a$ , in µm) of the samples. The results from the IF-Profiler were used to calculate the average surface roughness according to Equation 1.

$$R_a = \frac{1}{L} \int_0^L |z(x)| dx \tag{1}$$

Where L denotes the evaluation length, z the height and x the distance along the measurement.

## 2.2 Computational Set-up

The simulation process began with the creation of a cylinder with varying degrees of roughness by means of the ABAQUS software and the importation of the IGES file through the Gambit pre-processor, Figure 2. Following that, the complete cylinder model with wind tunnel was drawn. The next step involved the generation of the computational grid of a cylinder with roughness, where the size of the computational model was one third that of the real model used in the experiment. The selected surface and volume of the model were then meshed before the initial boundary specified the continuum types. The model was then exported into the FLUENT solver in an 'msh' format.



Figure 2. Flow chart CFD and numerical procedures.

The next step in the FLUENT was to define the solver, viscous and energy model of the study and the material properties which then set values that closely resembled the experimental conditions. Then, the boundary condition values, that were the same as the values used in the experiment, were fixed. To simplify the calculation process, the solution controls had to be adjusted by ascertaining the under-relaxation factors and discretization. In the case of rotating cylinder, the rotation velocity was prescribed on the cylinder wall. The value of non-rotation and rotation rate is defined on the moving cylinder wall in range 400 rpm. Before the iteration of the whole modelling started, the number of iterations had to be fixed. Once the iteration was completed and a converged solution had been achieved, the results were displayed and analysis of the post-processing tasks could be carried out.

The computational method involved the use of basic leading equations to model the problem domain. The physics of the flow was solved by the application of continuity and momentum equations. A commercial computational fluid dynamics package known as FLUENT was used to simulate the flow over a rotating cylinder with various degrees of surface roughness. The geometry of the cylinder was assigned a R<sub>a</sub> value to denote the surface roughness. The surface roughness of the cylinder was represented as a collection of tiny step function profiles throughout its surface based on a simulation of the actual cylinder surface profile, as can be seen in Figure 3. However, this method was restricted in the sense that it could not model the 'Finer Irregularities' on the surface of the cylinder. One way to bypass this limitation was to get an estimation of the actual rough surface on the geometry

by using the average height ( $R_a$ ) of the irregular surface<sup>11</sup>. The size of the irregular roughness was in micron meters ( $\mu$ m).

real rough surface (measured using Alicona IF-Profiler )



Approximation real rough

Figure 3. Principal schemes of roughness.

Simulations at various Reynolds number based on the same cylinder configuration in the experimental set-up are used to predict the forces around the rotating cylinder  $(2.0 \times 10^3 \le \text{Re} \le 4.2 \times 10^4)$ . With the cylinder centre at x = y = 0, the simulation confirms  $-0.7 \le x/D \le 1.7$  and  $-0.3 \le y/D \le 0.3$ , where x and y denote respectively the stream wise and transverse directions. At the velocity inlet on the left, uniform inflow (Ui) enters and then exists via the pressure outlet at the right, Figure 4. In the computational study, steady incompressible RANS equations are solved using implicit and segregated methods. The convective terms and viscous terms are discretised by second order upwind and central discretisation techniques respectively<sup>4</sup>. The simple algorithm is used for coupling of the pressure and velocity.



Figure 4. Domain and grid mesh.

Several blocks comprise the flow domain with multiple mesh densities to facilitate capture of local flow resolution at large velocity gradients. Micron-sized spacing and sizing is used in the design of the mesh points and surface roughness profile respectively, interspersed along the cylinder designed in ABAQUS software then converted to gambit files. Commercial mesh generation package GAMBIT is used in the mesh generation. The simulation is based on a typical structured mesh. The y + value near the cylinder surface is close to 1 to capture the effect of surface roughness.

The transported variable,  $\tilde{\nu}$  in the Spalart-Allmaras model was exactly the same as the turbulent kinematic viscosity except in the near-wall (viscous-affected) area. According to FLUENT<sup>4</sup>, the transport equation for  $\tilde{\nu}$  is:

$$\frac{d}{dt} \int_{V} \rho \Phi dv + \int_{dv} \rho \Phi \left( \vec{u} - \vec{u}_{s} \right) dA = \int_{dv} \Gamma \nabla \Phi dA + \int_{V} S_{\Phi} dV$$
(2)

Where  $G_{\nu}$  is the production of Turbulent Viscosity and  $Y_{\nu}$  is the destruction of Turbulent Viscosity that takes place in the near-wall area because of wall blocking and viscous damping, *and C* are constants,  $\nu$  is the molecular kinematic viscosity,  $S_{\nu}$  is a user-defined source term, and  $\mu$  is the Dynamic Viscosity. The Turbulent Viscosity was calculated from

$$\mu_t = \rho \tilde{\upsilon} f_{\nu 1} \tag{3}$$

Where  $f_{v1}$  is viscous damping function, which was expressed as

$$f_{\nu_1} = \frac{X^3}{X^3 + C_{\nu_1}^3} \tag{4}$$

Where  $C_{\nu 1}^3$  is the modelled constant, and

$$X = \frac{\tilde{v}}{v} \tag{5}$$

An integral form of the transport equation for a general scalar ( $\Phi$ ) on an arbitrary control volume (V) for a moving mesh is given as;

$$\frac{d}{dt} \int_{V} \rho \Phi dv + \int_{dv} \rho \Phi \left( \vec{u} - \vec{u}_{g} \right) dA = \int_{dv} \Gamma \nabla \Phi dA + \int_{V} S_{\Phi} dV$$
(6)

Where  $\vec{u}$  is the flow velocity vector, and  $\vec{u}_g$  is the grid velocity of the moving meshes. The first term on the left is the time derivative term while the second term is the convective term. Meanwhile, the first term on the right is the diffusive term, while the second term is the source term. The term  $\Gamma$  denotes the diffusion coefficient and  $S_{\Phi}$  denotes the source term ( $\Phi$ ). The term dV represents the boundary of V and dA as the area of movement.

## 3. Results and Discussion

#### 3.1 CFD versus Experiment

Figure 5 contains the results for Cylinder D, showing

the variations in C<sub>D</sub> due to the change of Reynolds number. Cylinder D measurements were taken in non-rotating and rotating conditions, with surface roughness. This particular set of parameters was selected because Cylinder D had the most roughness applied to the cylinder surface. Generally, in the non-rotating and rotating cylinder the drag decreases abruptly when the Re is increasing at  $\text{Re} < 1.0^4$ , and this is superseded by steadily increased drag between  $1.0^4$  < Re <  $1.4 \times 10^4$ . There is a final decreasing and tapering out for Re > 1.4x 10<sup>4</sup>. For  $1.0^4 < \text{Re} < 1.4 \text{ x } 10^4$ , C<sub>D</sub> increases with higher rotational speed. However, between  $1.4 \ge 10^4 < \text{Re} < 2.0 \ge$ 10<sup>4</sup>, there is a very minor effect on the drag coefficients, as it is assumed the inertial effects gain in dominance and cancel out the vertical effects. Beyond Re > 2.0 x10<sup>4</sup>, increased rotational speed appears to reduce the drag coefficients slightly, perhaps due to larger inertial effects which cause a delay in flow separation behind the cylinder, which has the result of decreasing the pressure drag (and overall  $C_D$ ) on the cylinder. It is worthy of note that at  $\text{Re} > 2.0 \text{x} 10^4$  the  $C_{D}$  predicted by CFD collapses into almost single data, regardless of rotational speed, presumed caused by limitations in the turbulence model and surface roughness modelling of the cylinder. The present validation of measurements for Cylinder D show overall agreement with the CFD analysis of the  $C_{D}$  for rotating cylinders with surface roughness, the average error being calculated at an acceptable level of < 10%.



**Figure 5.**  $C_{\rm D}$  comparison for non-rotating and rotating cylinder for Cylinder D, experimental versus CFD simulation.

### **3.2 Velocity Profiles**

The flow visualization was carried at Re < 2.0 x  $10^4$ 

and Re > 2.0 x 10<sup>4</sup> respectively. A noticeable at Figure 6 and Figure 7 was observed in stagnation point for Counter-Rotating Cylinder with Roughness at 400rpm. An estimated position of the flow from the upstream stagnation point has been enlarging with a small contour frame. The shape of the stagnation point for Re < 2.0 x  $10^4$  is skewed on the top direction when counter-rotating speed is 400rpm. However, at Re > 2.0 x  $10^4$  case is showed a complete parabolic shape of stagnation point. The difference shape of stagnation point is effecting from the high Counter-Rotating Speed (400 rpm) at low and high Reynolds number. This phenomenon i.e. in stagnation point occurring on the surface of rough cylinder is mainly responsible for starting flow visualization through the cylinders.







**Figure 7.** Stagnation point for counter-rotating cylinder with roughness at High Reynolds number.

Figure 8 shows the separation point has been occurring when the  $C_p$  is unstable. The angle at which the separation occurs is known as separation angle. The separation point is observed has asymmetrical separation at upper and lower surface. The vectors of velocity magnitude for separation point at the upper zone has been enlarge in Figure 8. Delay in separation process occurring on the surface of the cylinder mainly responsible for the reduction in drag coefficient of the Cylinders<sup>2</sup>.



**Figure 8.** Separation point for counter-rotating cylinder with roughness.

# 4. Conclusion

The effect on flow of a Counter-Rotating Circular Cylinder with Surface Roughness has been investigated. Velocity and the intrinsic nature of flow patterns are investigated for a range of Reynolds number and speed of rotation.

The flow visualization show comparable behaviours in the computational approaches. A two-dimensional method is very predictive of turbulent flow, due to faster convergence. However, the potential of the method has not been fully investigated, as this study is restricted to velocity magnitude and flow visualization. More expansive studies, to include aerodynamics force, would be beneficial in verifying the applicability of this method.

Surface Roughness causes a delay in flow separation in the near-wake of a rotating cylinder, which decreases pressure difference across the areas immediately in front and behind the cylinder. From this the conclusion reached is that drag coefficients are lower when there is surface roughness.

## 5. Acknowledgment

Extracts contained in this paper originate from studies that took place at University Sains Malaysia, sponsored by the Ministry of Higher Education. The research is also co-sponsored by Universiti Teknologi MARA under the Fundamental Research Grant Scheme (600-RMI/FRGS 5/3 (144/2014)). Particular gratitude is extended to our Department - Mechanical Engineering Department, Polytechnic Seberang Perai.

## 6. References

- 1. Achenbach E. Influence of Surface Roughness on the Crossflow Around a Circular Cylinder. Journal of Fluid Mechanics. 1971; 46(02):321–35.
- Butt U, Egbers C. Aerodynamic Characteristics of Flow over Circular Cylinders with Patterned Surface. International Joyrnal of Materials, Mechanics and Manufacturing. 2013; 1(2):121–25.
- 3. Chng T et al. Flow Control of an Airfoil via Injection and Suction. Journal of Aircraft. 2009; 46(1):291–300.
- 4. Fluent. Fluent 6.3 Documentation; 2006.
- Fransson JH, Konieczny P, Alfredsson PH. Flow Around a Porous Cylinder Subject to Continuous Suction or Blowing. Journal of Fluids and Structures. 2004; 19(8):1031–48.
- Gbadebo SA, Cumpsty NA, Hynes TP. Control of Three-dimensional Separations in Axial Compressors by Tailored Boundary Layer Suction. Journal of Turbomachinery. 2008; 130(1):011004.
- Grager T, Rothmayer A, Hu H. Stall Suppression of a Low-Reynolds-Number Airfoil with a Dynamic Burst Control Plate. In 49th AIAA Aerospace Sciences Meeting Including the New Horizons Forum and Aerospace Exposition, AIAA; 2011.
- 8. Greenblatt D et al. Experimental Investigation of Separation Control part 1: Baseline and Steady Suction. AIAA Journal. 2006; 44(12):2820–30.
- 9. Gu M, Du X. Experimental Investigation of Rain-Wind-

Induced Vibration of Cables in Cable-Stayed Bridges and its Mitigation. Journal of Wind Engineering and Industrial Aerodynamics. 2005; 93(1):79–95.

- Hikami Y, Shiraishi N. Rain-wind induced vibrations of cables stayed bridges. Journal of Wind Engineering and Industrial Aerodynamics. 1988; 29(1):409–18.
- 11. Hopkins PE et al. Effects of Surface Roughness and Oxide Layer on the Thermal Boundary Conductance at Aluminum/silicon Interfaces. In 2010 14th International Heat Transfer Conference; American Society of Mechanical Engineers; 2010.
- Li H, et al. Vibration Control of Stay Cables of the Shandong Binzhou Yellow River Highway Bridge using Magnetorheological Fluid Dampers. Journal of Bridge Engineering. 2007; 12(4):401–09.
- Main JA, Jones NP. A Comparison of Full-scale Measurements of Stay Cable Vibration. In Proceedings of Structures Congress 2000.
- 14. MatsumotoM, et al. Field observation of the Full-scale Wind-induced Cable Vibration. Journal of Wind Engineering and Industrial Aerodynamics. 2003; 91(1):13–26.
- Owen JC, Bearman PW, Szewczyk AA. Passive control of VIV with Drag Reduction. Journal of Fluids and Structures. 2001; 15(3):597–605.
- 16. Patnaik B, Wei G. Controlling Wake Turbulence. Physical Review Letters. 2002; 88(5):054502.
- 17. Patil RS, Ng TT. Control of Separation using Spanwise Periodic Porosity. AIAA Journal. 2010; 48(1):174–87.
- Wu C-J, Wang L, Wu J-Z. Suppression of the Von Karman Vortex Street behind a Circular Cylinder by a Travelling Wave generated by a Flexible Surface. Journal of Fluid Mechanics. 2007; 574:365–91.
- Zuo D, Jones N, Main J. Field observation of Vortex-and Rain-Wind-Induced Stay-Cable Vibrations in a Three-Dimensional Environment. Journal of Wind Engineering and Industrial Aerodynamics, 2008; 96(6):1124–33.
- Zuo D, Jones NP. Interpretation of Field Observations of Wind-and Rain-Wind-Induced Stay Cable Vibrations. Journal of Wind Engineering and Industrial Aerodynamics. 2010; 98(2):73–87.